COMPUTATIONAL STUDY OF TURBINE BLADE FILM COOLING PERFORMANCE USING ANISOTROPIC TURBULENCE MODELS

CHANDRAN SUNDARAJ

A thesis submitted in fulfilment of the
requirements for the award of the degree of
Master of Engineering (Mechanical)

Faculty of Mechanical Engineering
Universiti Teknologi Malaysia

MARCH 2006

ACKNOWLEDGEMENT

The completion of this work was possible with the support of many individuals. Although it is not possible to list all of them here, I would like to mention a few who deserve my deepest appreciation.

I wish to express my sincere gratitude to my thesis supervisors, Assoc Prof Dr. Tholudin and Dr. Jamaluddin, for their guidance, support and advice throughout my research.

I would also like to thank my parents, sisters and brothers for their love and support to pursue my graduate studies.

I am very grateful to Mr. Muthu and family for their caring, support and encouragement throughout my studies. Their financial support really helps me to finish this study.

I must thank my fiancée, Miss Letchumy Praba for her invaluable support and encouragement. She has always been with me during good and bad times.

ABSTRACT

Three dimensional low Mach number film cooling of turbine blade have been conducted using computational fluid dynamics (CFD) software FLUENT. Strong anisotropic of film cooling turbulence and flow complexities require capable turbulence model such as Reynolds Stress Model (RSM) or Large Eddy Simulation (LES) model to solve film cooling flow field. Film cooling with holes arrangement on blade leading edge, pressure and suction were tested in present study. The effects of film cooling parameters such as blowing ratio, surface curvature, injection angle, hole spacing, hole length, and plenum geometry have been investigated. The results presented in adiabatic film cooling effectiveness as well as plots of temperature and velocity contour. Present study reveals that blowing ratio, injection angles and coolant holes arrangements are significant parameters in film cooling process. Performances of film cooling highly depend on a combination of parameters. Present study represents the feasibility of CFD utilization as an innovative predictive tool in turbine blade film cooling design.

ABSTRAK

Aliran penyejukan saput bilah turbin tiga dimensi dengan nombor Mach yang rendah di jalankan menggunakan perisian pergerakan bendalir berkomputer, FLUENT. Aliran gelora penyejukan saput yang tidak terarah dan struktur aliran yang komplek memerlukan model aliran gelora yang berkebolehan seperti Reynolds Stress Model (RSM) atau Large Eddy Simulation (LES) untuk penyelesaian. Penyejukan saput dengan aturan lubang pada pinggir depan, permukaan cembung dan cekung bilah di kaji. Kesan parameter penyejukan saput seperti nisbah pancutan, kelengkungan permukaan, sudut pancutan, jeda lubang, panjang saluran bahan penyejuk, bentuk lubang dan geometri ruang pembekal penyejuk turut dikaji. Keputusan kajian dipersembahkan dalam bentuk kecekapan adiabatik penyejukan saput serta melalui plot kontor suhu dan halaju.Perbincangan keputusan adalah berdasarkan pengaruh parameter ini ke atas struktur aliran penyejukan saput. Kajian ini memdedahkan bahawa nisbah pancutan, sudut pancutan dan susunan lubang mempunyai kesan yang ketara keatas kecekapan penyejukan saput.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS

b	-	Constant in the turbulence transport equations
C_p	-	Specific heat at constant pressure, (kJ/kg.K)
C_μ	-	Empirical constant or variable in the turbulence equations
D	-	Coolant Hole Diameter, (m)
DR	-	Density ratio, ($DR=\rho_j/\rho_\infty$)
E	-	Total Energy, (kJ)
f	-	Mixture fraction
G_k, G_b	-	Generations terms in the turbulence transport equations
h	-	Local heat transfer coefficient, (W/m^2. K)
J	-	Mass flux, Diffusion Flux (kg/m^2-s)
K	-	Thermal conductivity, (W/m.K)
k	-	Turbulent kinetic energy, (m^2/s^2)
k	-	Mass Transfer Coefficient
L	-	Length of coolant hole, (m)
M	-	Blowing ratio, ($\rho_j U_j/\rho_\infty U_\infty$)
m	-	Mass (kg)
P	-	Pressure, (N/m^2);also refers to first element of the wall
p	-	Spanwise distance/pitch between coolant holes centerline (m)
q''	-	Local heat flux at the wall, (W/m^2)
Re	-	Reynolds number, ($\rho UD/\mu$)
S	-	Modulus of the mean rate of strain tensor in the turbulence transport equations
S_ε, S_k	-	Source term in the turbulence transport equations
S_{ij}	-	Rate of strain tensor in the Reynolds stress tensor
T	-	Temperature, (K)
T_{aw}	-	Adiabatic wall temperature, (K)

Ti	-	Turbulence Intensity
U	-	Averaged velocity components, (m/s)
$\overline{u_i' u_j'}$	-	Reynolds stress tensor
u_τ	-	Frictional velocity, ($\sqrt{\frac{\tau_w}{\rho_w}}$)
x	-	Streamwise distance measured from coolant hole center, (m)
y	-	Vertical distance measured blade surface, (m)
y^+	-	Non dimensional wall distance, ($\sqrt{\frac{\rho u_\tau y_p}{\mu}}$)
y_p	-	Height of the first node P off the wall, (m)
z	-	Lateral distance /span wise of the blade (m)
α	-	Injection / inclination angle of coolant jets
α_k , α_ε	-	Inverse effective Prandtl numbers for k and ε in the turbulence transport equations
β	-	Orientation angle of coolant jets.
δ_{ij}	-	Kronecker delta,
ε	-	Dissipation rate of turbulent kinetic energy, (m^2/s^3)
η	-	Local adiabatic film cooling effectiveness, ($\frac{T_{aw} - T_\infty}{T_j - T_\infty}$)
$\bar{\eta}$	-	Spanwise averaged film cooling effectiveness, ($\frac{\overline{T}_{aw} - T_\infty}{T_j - T_\infty}$)
μ	-	Laminar dynamic viscosity, (kg/m.s)
μ_T	-	Turbulent dynamic viscosity (kg/m.s)
υ	-	Laminar kinematic viscosity, (s/m^2)
ρ	-	Density, (kg/m^3)
σ_k, σ_ε	-	Turbulent Prandtl numbers for k and ε in the turbulence transport equations
τ	-	Shear stress, (N/m^2)

CHAPTER 1

INTRODUCTION

1.1 Problem Statement

The needs of protection of a surface due to extreme thermal condition by means of a fluid film were applied in many applications. Harsh and extreme thermal environment such as inside combustion chamber and turbine blade can lead to failure of their surface walls. The continuous improvement in the performance of gas turbine engine necessitates a continuous increase in the turbine inlet temperature. The efficiency and power output of gas turbine engine increase with increment in turbine inlet temperature. Is estimated that for every 100 ^{0}F increase in this temperature, the power output increases by about 10% and the efficiency increases by about 1.5% Berhe and Patankar (1998).

Since the inlet temperature of present generation gas turbines are much higher than the melting temperature of the available alloys used to make turbine blades. Hence cooling of the turbine blades is a critical issue in gas turbine engine technology. A suitable cooling mechanism is needed in term to reduce the resulting high thermal stresses and to prolong the life of turbine blade under this extreme condition. Currently, turbine blades are cooled by a combination of internal and external cooling mechanism. Convection and impingement cooling are the methods used to cool the turbine blade internally.

Film cooling is most common cooling mechanism for external surface of turbine blades. Film cooling wherein cooler air from the compressor is bled thru cavities of connecting shaft and injected near the blade surface through holes or slots to provide a layer of cool fluid between the hot gaseous and the blade surface. The

objective of film cooling is to provide a blanket of cold film, which behaves as an insulation layer on blade surface or as a heat sink.

Film cooling on turbine blade is controlled by many variables that contribute significantly on cooling effectiveness and local heat transfer coefficient over blade surface. Figure 1.1 depicted the turbine blade film cooling and its flow structures. Film cooling mainly affected by the blowing ratio, surfaces curvature, the mainstream turbulence intensity, the holes injection and compound angle, the holes spacing and the hole geometry. Recent studies also highlighted the importance of coolant supply hole and plenum geometry film cooling application. Numbers of the film cooling research of the past 30 years has been conducted especially on flat plat. Goldstein (1971), Simoneau and Simon (1993) and Bunker R.S (2005) provided intensive reviews on film cooling research. Although many important explanations have been obtained from these studies such as the qualitative understanding of the effects of a number of film cooling variables the quantitative applicability of these explanations to cooling of actual turbine blades is much less known.

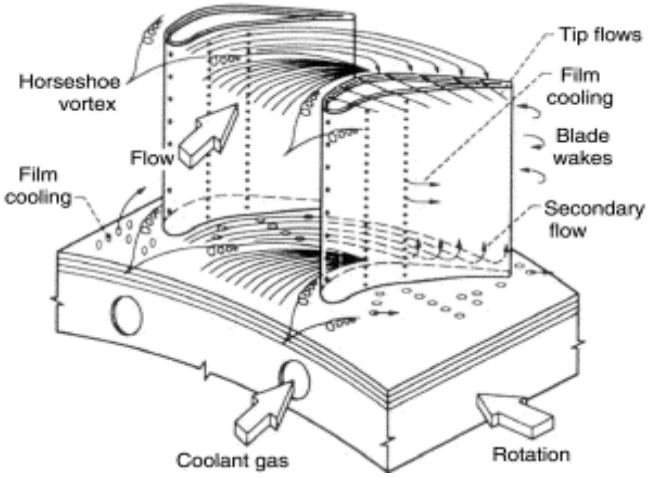

Figure 1.1 General schematic of turbine blade film cooling and flow structures (Garg, V. K., 2002)

In film cooling application, especially on turbine vane and blade there are certain requirements must be considered. First, the coolant quantity should be the minimum possible. This is due to excessive coolant gases into the mainstream undercuts the production of useful power; this is because the coolant air that is taken from compression would go to produce real power. Moreover increased coolant

quantity will require more force to delivery it. Large quantity of coolant will interfere with the normal functioning of combustion gaseous and will reduce the gas turbine efficiency. Second, jet penetration into mainstream should be minimised. If jet penetration into mainstream is high, the coolant gases are lost into the mainstream instead of protecting the blade surfaces. Third, a good lateral spread of the coolant gases is important in order to provide uniform coolant coverage over the blade. Finally, the disruption to the blade aerodynamics must be minimum.

Film cooling studies on turbine by experimental test are very expansive and time consuming. Computational method in other hand gained popularity as alternative tool in current year. Computational fluid dynamic (CFD) can be a good and affordable tool when experimental test is impossible. Furthermore increase of storage capacity and computation speed may able to simulate complex flow problem with high accuracy and less cost compare to experimental test. This study proposes to investigate the turbine blade film cooling by computational method. Through computational fluid dynamic (CFD) the behaviour of turbine blade film cooling and its affecting parameters will be investigated. It is believed this work will provide a good contribution in understanding turbine blade film cooling.

1.2 Objective

The goal of this study is to use numerical method to investigate the turbine blade film cooling effectiveness and its parameters.

1.3 Scope of Research

In this study an investigation has been conducted to determine the effects of several film cooling parameters on cooling performance using commercial computational fluid dynamic (CFD) software, FLUENT. The blowing ratio, surface curvature, free stream turbulence intensity, coolant injection angle, compound angle

and holes spacing were investigated. The computational domain consists of main flow region, coolant hole and supply plenum placed at blade leading edge, suction and pressure surface. The main scope of this study directed as follows:

1) Benchmark studies and validation of computational aspects in film cooling application.
2) Investigate the ability of the computational model to accurately predict the film cooling application.
3) Simulate low Mach number three-dimensional turbine blade film cooling using anisotropic turbulence model.
4) Discus and analyse effect of the film cooling parameters using computational data.

1.4 Outline of Thesis

Chapter 2 of this thesis provide a literature reviews on film cooling research. This chapter start with brief review on film cooling theory and fundamentals and followed by reviews on computational studies on film cooling. Film cooling flow structures discussed in following section and then various film cooling parameters effects were discussed.

Research methodology been discussed in Chapter 3. Various computation aspects such as computation model, grid generation, governing equations, and turbulences models explained. FLUENT CFD components also reviewed in this chapter. In Chapter 4, results of validation and benchmark solution of flat plat film cooling and cylindrical leading edge film cooling presented and discussed. Grid independence and turbulences models been assessed to attain the yardstick for present simulation.

In Chapter 5 results and discussions on turbine blade film cooling are presented. Anisotropic turbulences models, RSM and LES utilised to conduct the simulation and both models were compared accordingly. Blade leading edge, suction and pressure surface film cooling investigated using temperature and velocity plots.

Film cooling adiabatic effectiveness calculated at stream direction and blade spans. Parametric variations on blade film cooling also presented in this chapter. The conclusion and recommendations for future work presented in Chapter 6.